# ETABS®

## Integrated Building Design Software

## Introductory User's Guide

**Computers and Structures, Inc.**
**Berkeley, California, USA**

Version 8
February 2003

# Copyright

The computer program ETABS and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

Phone: (510) 845-2177
FAX: (510) 845-4096
e-mail: info@csiberkeley.com (for general questions)
e-mail: support@csiberkeley.com (for technical support questions)
web: www.csiberkeley.com

© Copyright Computers and Structures, Inc., 1978-2003.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
ETABS is a registered trademark of Computers and Structures, Inc.
Windows is a registered trademark of Microsoft Corporation.
Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated.

# DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF ETABS. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

# Contents

## Introductory User's Guide

1 Program Description

2 ETABS “Screen”

3 Basic Modes, Drawing Tools, Mouse Pointers

4 Begin a Model

5 Create the Structural Model

6 Select Structural Objects

7 Load the Structural Model

8 Assign/Change Properties

9 Edit the Model Geometry

10 Analyze the Model

11 Design

12 Graphical Displays

13 Generate Results

Chapter 1

# Program Description

## Objective

This chapter briefly describes the program and some of the concepts involved in its use.

## This Is ETABS

ETABS is a powerful program that can greatly enhance an engineer's analysis and design capabilities for structures. Part of that power lies in an array of options and features. The other part lies in how simple it is to use.

The basic approach for using the program is very straightforward. The user establishes grid lines, places structural objects relative to the grid lines using points, lines and areas, and assigns loads and structural properties to those structural objects (for example, a line object can be assigned section properties; a point object can be assigned spring properties; an area object can be assigned slab or deck properties). Analysis and

design are then performed based on the structural objects and their assignments. Results are generated in graphical or tabular form that can be printed to a printer or to a file for use in other programs.

In using the program, you manage the **File**, **Edit** the model, change the **View**, **Define** properties or load cases, **Draw** something new in the model, **Select** that something, **Assign** properties or loads, **Analyze** the model, **Display** analysis results for checking, **Design** the structure, apply various **Options** to achieve the desired outcome with optimum effort, and seek **Help** when you need it. Those actions are the basis for the program menu structure. Thus, familiarity with the menu commands and their function is key to expanding your ability to use ETABS.

***ETABS Menu Commands:***

- *File*
- *Edit*
- *View*
- *Define*
- *Draw*
- *Select*
- *Assign*
- *Analyze*
- *Display*
- *Design*
- *Options*
- *Help*

***Note:***

*The ETABS Software Verification Manual documents analysis using ETABS.*

Information about the various menu items is available using the **Help menu > Search for Help on** command as well as by using the F1 key when a form is displayed on the ETABS screen. The F1 key will display context sensitive help, including descriptions of the types of input for the forms used in the program. Familiarity with the menu commands will enable the user to create models for complex Composite Floor Framing Systems with Openings and Overhangs, Steel Joist Systems, Moment Resisting Frames, Complex Shear Wall Systems, Rigid and Flexible Floors, Sloped Roofs, Ramps and Parking Structures, Mezzanine Floors, Trussed Systems, Multiple Tower Buildings and Stepped Diaphragm Systems, and many more.

Technical Notes in .pdf format are available using the **Help menu > Documentation and Tutorials** command. Those Notes explain how the program performs concrete frame design, steel frame design, composite floor design, steel joist design, and concrete shear wall design in accordance with applicable building codes.

# Time Saving Options

The program also includes options that allow you to reduce the time spent creating models. Those options include the following:

- **Similar Stories.** Allows the user to make changes to multiple stories simultaneously.
- **Snap To.** Allows the user to place structural elements with accuracy.
- **Auto Select Sections.** Allows the user to define a list of sections, for example W18X35, W18X40, W21X44, W21X50 and W24X55, that can be assigned to a frame member. The program can then automatically select the most economical, adequate section from the auto select section list when it is designing the member.
- **Vertical Load Transfer.** Frees the user from the chore of calculating the load on the members supporting the floor plate, and determines the area tributary to each member for live load reduction.

# Templates and Defaults

ETABS provides a number of templates that allow for the rapid generation of models for a wide range of common building types. Those templates serve as a good starting point because they can be modified easily.

The program also includes templates for two-dimensional and three-dimensional frames that can be appended to an existing model. The two-dimensional option can be used to locate planar frames throughout a model. The three-dimensional option can assist in modeling conditions where several towers rest on the same base structure.

The program includes defaults parameters, many of which are building code specific. Those defaults are accessed using "Overwrites" and "Preferences." The possible options available for overwrites and the default values for preferences are identified in the design manuals.

By using the built-in templates and defaults, the user can create a model in a matter of minutes.

1

# Basic Process

The following provides a broad overview of the basic modeling, analysis, and design processes:

1. Set the units.
2. Open a file.
3. Set up grid lines.
4. Define story levels.
5. Draw structural objects.
6. Define frame properties.
7. Define loads.
8. Edit the model geometry.
9. Assign properties.
10. View the model.
11. Analyze the model.
12. Display results for checking.
13. Design the model.
14. Generate output.
15. Save the model.

# Forms

Various forms are used in ETABS throughout the modeling, analysis and design processes. With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

Chapter 2

# ETABS “Screen”

## Objective

This chapter briefly describes the ETABS “screen” or more accurately, the graphical user interface.

## The ETABS Window

The ETABS graphical user interface shown in Figure 2-1 includes the main window, main title bar, display title bar, menu bar, toolbars, display windows, status bar, mouse pointer position coordinates and the current units. Each of these items is described in the bulleted list that follows.

- **Main Window.** This window may be moved, resized, maximized, minimized, or closed using standard Windows operations. Refer to Windows help, available on the Start menu, for additional information about those items.

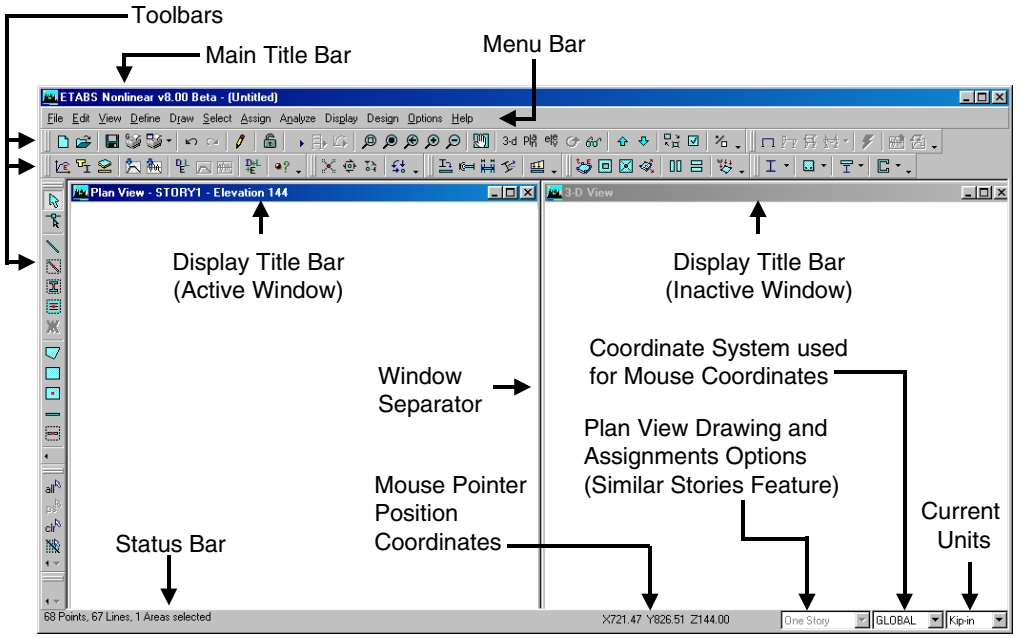



**Figure 2-1: The ETABS graphical user interface**

- **Main Title Bar.** The main title bar includes the program and model names. The main title bar is highlighted when the program is in use. Move the main window by left clicking in the main title bar and holding down the mouse button as you drag the window around your screen.

- **Menu Bar.** The menu bar contains all of the program's menus.

- **Toolbars and Buttons.** Toolbar are made up of buttons. Buttons provide "one-click" access to commonly used commands. Holding the mouse pointer over a toolbar button for a few seconds without clicking or holding down any mouse buttons will display a short description of the button's function in a small text box.

- **Display Windows.** A display window shows the geometry of the model and may also include displays of properties, loading and analysis or design results. Up to four windows may display at any one time.

- **Display Title Bar.** The display title bar is located at the top of the display window. The display title bar is highlighted when the associated display window is active. The text in the display title bar typically includes the type and location of the view in the associated display window.

- **Status Bar.** The status bar is located at the bottom of the main window. Text describing the current status of the program is displayed on the left side of the status bar.

- **Mouse Pointer Position Coordinates.** The mouse pointer position coordinates are displayed on the right-hand side of the status bar. A window does not need to be active for the mouse pointer position coordinates to be displayed. It is only necessary that the mouse pointer be over the window. In a two-dimensional plan or elevation view, the mouse pointer position coordinates are always displayed. In a three-dimensional view, the mouse pointer position coordinates are only displayed when the mouse pointer snaps to a point or a grid line intersection.

- **"One Story" Drop-Down Box.** This drop-down box is on the right side of the status bar. The three options in the drop-down box are One Story, All Stories, and Similar Stories. With One Story, an object is applied only to the story level on which it is drawn. With All Stories, an object drawn in the plan view is applied to all story levels in the model at the same plan location. An assignment made to the selected objects also is made to the other objects in the same plan location at all other story levels. With Similar Stories, an object drawn in plan view is applied to all similar story levels in the model at the same plan location. An assignment made to the selected objects is made to the other objects in the same plan location at all similar story levels.

- **Current Units.** The current units are displayed in a drop-down box located on the far right-hand side of the status bar. You can change the units at any time during the model creation process.

2

# The Aerial View

Figure 2-2 shows an example of the aerial view window. This window displays the entire drawing to help you move around the active window of a larger model and use the zoom feature to view smaller areas more easily. Also use the aerial view to track which part of the model is displayed in the active window. Each time the model is edited, the aerial view is updated.

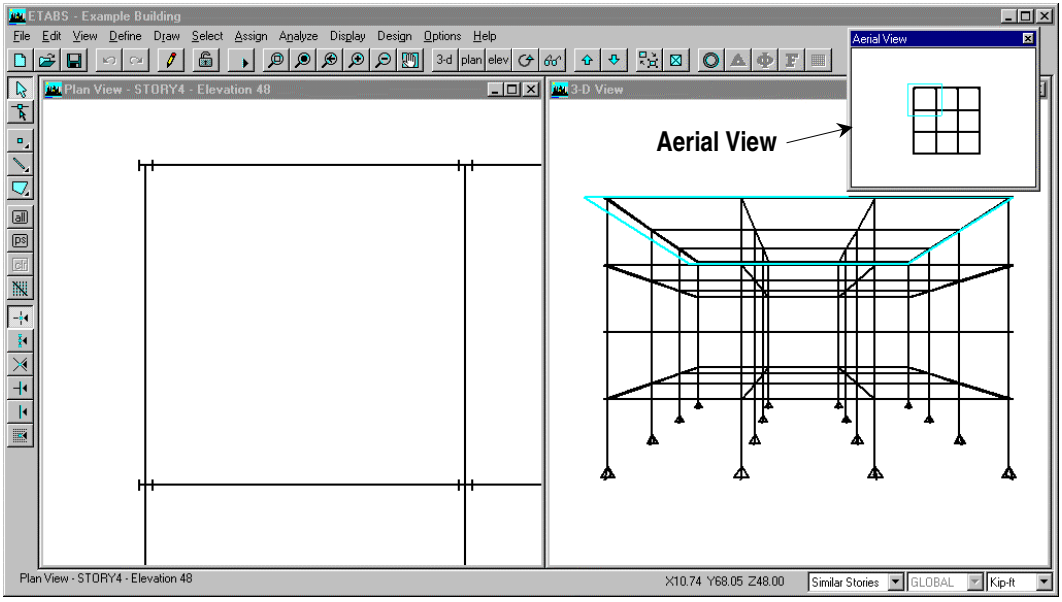



**Figure 2-2: Example of the aerial view**

Chapter 3

# Basic Modes, Drawing Tools, Mouse Pointers

## Objective

This chapter briefly describes the two modes of user operation for ETABS, identifies the drawing tools, and describes how the appearance of the mouse pointer changes for various operations.

## Select or Draw

The two distinct modes in this program are the ***select*** mode and the ***draw*** mode.

- The select mode allows you to select objects and is used for editing operations, making assignments to objects, and viewing or printing results. By default, the program is in select mode. Chapter 6 describes the various methods for selecting points, lines, and areas in your model.
- The draw mode allows you to draw objects.

The draw mode automatically enables when you select one of the following submenu options from the Draw menu or click on the corresponding buttons on the toolbar. Note that the views in parenthesis (Plan, Elev, 3D) after the command name indicate when the button will be active; for example, the Draw Lines command/button can be used in the Plan, Elevation or 3D views, but the Draw Walls command/button can be used only in Plan view. The names of the commands are assumed to explain the actions that will be accomplished. The terminology "in Regions" means within a bay and "at Clicks" means at the location of the mouse pointer in the model when you click the left mouse button. More information about the Draw tools is available by searching for "draw menu" using the **Help menu > Search for Help on** command.

- Draw Point Objects
- Draw Line Objects
  - Draw Lines (Plan, Elev, 3D)
  - Create Lines at Regions or at Clicks (Plan, Elev, 3D)
  - Create Columns in Regions or at Clicks (Plan)
  - Create Secondary Beams in Regions or at Clicks (Plan)
  - Create Braces in Regions or at Clicks (Plan)
- Draw Area Objects
  - Draw Areas (Plan, Elev, 3D)
  - Draw Rectangular Areas (Plan, Elev)
  - Create Areas at Click (Plan, Elev)
  - Draw Walls (Plan)
  - Create Walls in Regions or at Clicks (Plan)
- Draw Developed Elevation Definition

- Draw Section Cut
- Draw Dimension Line
- Draw Reference Point

The draw mode remains enabled until you do one of the following to return to the select mode:

- Click the Pointer button on the toolbar .
- Press the Esc key on the keyboard.
- Select a command from the Select menu.

**The mouse pointer indicates which mode is enabled.** The appearance/properties of the mouse pointer are defined in the Windows Control Panel. The mouse pointer properties are Normal Select Pointer and Alternate Select pointer.

***Note:***

*Typically, set the properties for the mouse by clicking on the Windows Start menu, then Settings, then Control panel and clicking on Mouse to bring up your Mouse properties form.*

In select mode, the pointer is the Normal Select Pointer. If you are using the default settings, the mouse pointer will look like this .

In draw mode, the mouse pointer is the Alternate Select pointer. If you are using the default settings, the mouse pointer will look like this .

Note that while in draw mode, if you run the mouse pointer over the toolbar buttons or the menus, the pointer temporarily changes to the selection pointer. If you do not click on one of the menus or toolbar buttons, the mouse pointer reverts to the draw mode pointer when you move back into the display window.

Other mouse properties/appearances are used for various actions in the program, including Help Select, Busy, Text Select, Vertical Resize, Horizontal Resize, and Move. The appearance of the mouse pointers for those actions depends on the mouse pointer properties you specify.

Chapter 4

# Begin a Model

## Objective

This chapter describes how to begin a model by creating the basic grid system. Structural objects are placed relative to the grid system.

## Create the Basic Grid System

Begin creating the grid system by clicking the **File menu > New Model** command or the **New Model** button . The form shown in Figure 4-1 will display.

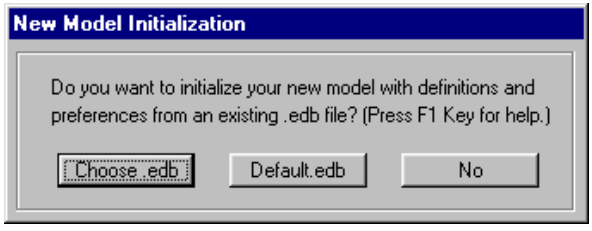

New Model Initialization
Do you want to initialize your new model with definitions and preferences from an existing .edb file? (Press F1 Key for help.)
Choose .edb
Default.edb
No

**Figure 4-1 The New Model Initialization form**

*Note:*

*More information about templates is available by searching for "template" using the* ***Help menu > Search for Help on*** *command..*

Select the **No** button on that form and the form shown in Figure 4-2 will display.

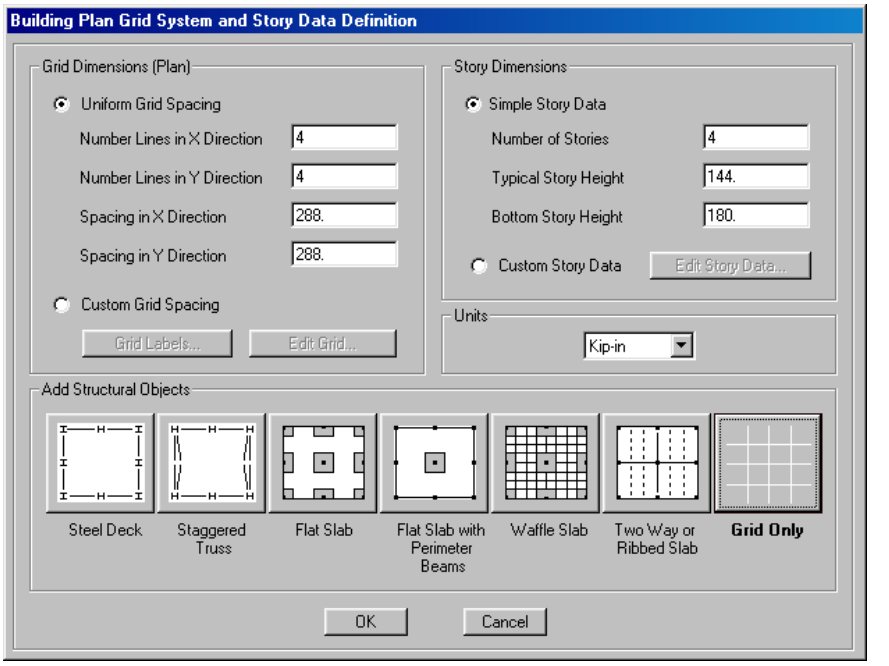



**Figure 4-2 Building Plan Grid System and Story Data Definition form**

The Building Plan Grid System and Story Data form is used to specify horizontal grid line spacing, story data, and, in some cases, template models. Template models provide a quick, easy way of starting your model. They automatically add structural objects with appropriate properties to your model. We highly recommend that you start your models using templates whenever possible. However, in this example, the model is built from scratch, rather than using a template.

The form has **OK** and **Cancel** buttons, which are used to accept or cancel the selections made on the form. Click the **OK** button for any selections/entries to be accepted. Clicking the **Cancel** button cancels any selections/entries.

## Grid Dimensions (Plan) - Define a Grid System

Use the Grid Dimensions (Plan) area of the form to define a grid line system. Select from two options for defining the grid line system:



- **Uniform Grid Spacing.** Specify the number of grid lines in the X and Y directions and a uniform spacing for those lines. Note that the uniform spacing in the X and Y directions can be different. This option defines a grid system for the global coordinate system only. If subsequently necessary, edit the information using the **Edit menu > Edit Grid Data** command. For more information, search for "edit grid data" using the **Help menu > Search for Help on** command. Note that the default global coordinate/grid system is a Cartesian (rectangular) coordinate system. Use the **Edit > Edit Grid Data > Edit Grid** command to modify the grid system.

- **Custom Grid Spacing.** Label grid lines and define nonuniformly spaced grid lines in the X and Y directions for the global coordinate system. After choosing this option, click the **Grid Label** button to label grid lines and click the **Edit Grid** button to edit the grid system. For more information, search for "grid labeling" using the **Help menu > Search for Help on** command.

The reasons for defining a grid system for the model include the following:

- Default elevation views in the model occur at each defined primary grid line in your model.

- Structural objects added to the model from a template are added based on the grid line definitions in the model.

- Objects snap to grid lines when drawn in the model.

- Objects mesh at their intersections with grid lines.

- The grid lines in the model can be defined with the same names as are used on the building plans. This may allow for easier identification of specific locations in the model.

Chapter 5

# Create the Structural Model

## Objective

This chapter describes how to create the structural model. It is assumed that you have read Chapter 4 *Begin a Model* or understand how to begin an ETABS model by defining a grid system.

## Define Story Data

***Note:***

*The story data of an existing model can be changed using the* ***Edit menu > Edit Story Data > Edit Story*** *command.*

Story data is defined using the Plan Grid System and Story Definition form. Figure 4-2 of Chapter 4 shows this form. As described in Chapter 4, use the **File menu > New Model** command and make a selection on the New Model Initialization form to bring up the Building Plan Grid System and Story Definition form. Select one of the two options in the Story Dimensions Area of the form to define the story data:

- **Simple Story Data:** Enter values in the edit boxes to define the number of stories and a typical story height that is used for all story levels.

*Note:*

*Story level "similarity" can be significant. For example, when Story 2 is "Similar To" Story 1, an object drawn on Story 1 typically appears in the same plan location on Story 2.*

The program provides default names for each story level (for example, Story 1, Story 2 and so on) and assumptions for story level similarity.

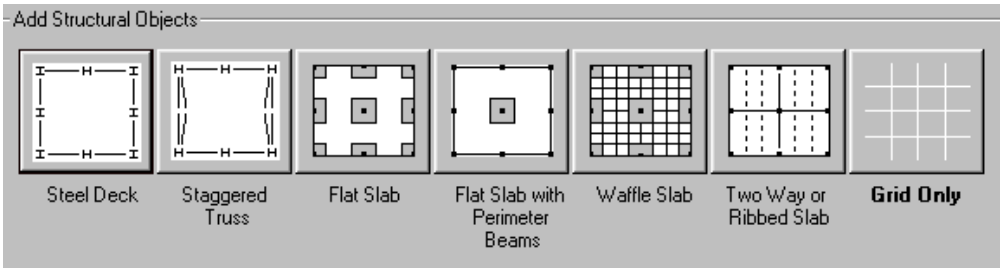
- **Custom Story Data:** After choosing this option, click the **Edit Story Data** button to bring up the Story Data form. Enter values in the Story Data form to define your own story names, story levels of non-uniform height and customized story similarity.

The Story Data form also appears when the **Edit menu > Edit Story Data > Edit Story** command is used. For more information about the Story Data form, refer to the section entitled "Edit Story Data Command" in the Edit Menu chapter of the graphical user interface manual. For more information about story level similarity, search for "similar stories drop-down box" using the **Help menu > Search for Help on** command. Story level similarity can also be significant to composite beam and steel joist design. Search for "similarity" using the **ETABS Help menu > Search for Help on** command for more information.

# Add Structural Objects Using Templates

Use the bottom half of the same Building Plan Grid System and Story Definition form as was used in the previous section to add structural objects to your model from one of several built-in templates. In many cases it is the simplest, most convenient and quickest way to start your model.

The Add Structural Objects area of the Building Plan Grid System and Story Definition form is reproduced herein for reference:



Note that the templates consist of two for steel building, four for concrete building, and one for grids only, which means that no structural objects are added to the model from the template. When an option (button) has been selected in the Add Structural Objects area, its name will be highlighted. When the Building Plan Grid System and Story Definition form opens, the Grid Only selection is highlighted, thus indicating that unless you select another template, your model will have only a grid system.

Choose any of the templates by left clicking its associated button. When one of the template buttons is chosen, a form for that template appears. Use the template form to specify various types of data for your model. When you have finished specifying data on the template form, click the **OK** button to return to the Building Plan Grid System and Story Definition form. Click the **OK** button on the Building Plan Grid System and Story Definition form to complete the operation.

**Note:** When using concrete building templates in this program, beams and slab ribs (joists) are normally modeled with depths equal to the dimension from the top of the slab (not bottom of slab) to the bottom of the beam or slab rib. Also, beams are modeled as line elements in this program. Thus, slabs with out-of-plane bending capability span from center-of-beam to center-of-beam in the program model.

After the **OK** button on the Building Plan Grid System and Story Definition form has been clicked, the model appears on screen in the main ETABS window with two view windows tiled vertically, a Plan View on the left and a 3-D View on the right, as shown in Figure 5-1. The number of view windows can be changed using the **Options menu > Windows** command.

Note that the Plan View is active in Figure 5-1. When the window is active, the display title bar is highlighted. Set a view active by clicking anywhere in the view window.

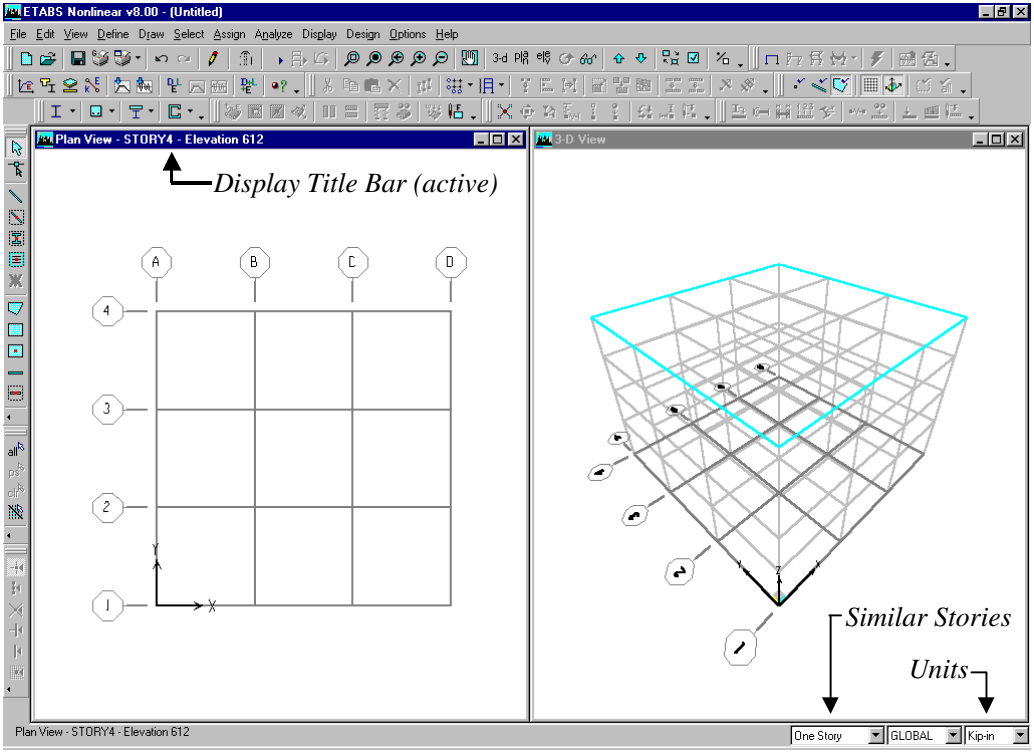



**Figure 5-1 The ETABS main window**

# Add Structural Objects Manually

Objects, such as columns, beams, and floors, also can be drawn manually as described in the sections that follow.

## Draw Columns

Make sure that the Plan View is active. Click the **Create Columns in Region or at Clicks** button or use the **Draw menu > Draw Line Objects > Create Columns in Region or at Clicks** command. The Properties of Object pop-up box for columns shown in Figure 5-2 will appear.

***Note:***

*The definition parameters and drawing controls available in the Properties of Object box differ depending on the drawing command/ button selected. Always check that the parameters and controls are what you need for the type of object you are drawing.*

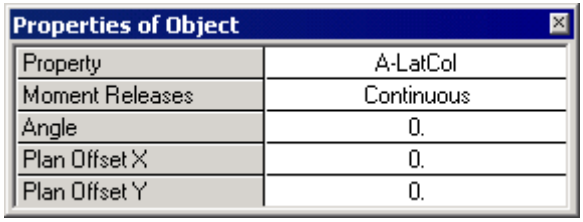



**Figure 5-2 Properties of Object Box for Columns**



The Properties of Object box provides various definition parameters and drawing controls. Review the parameters and controls shown in this box before drawing your column to ensure that they are what you want. Change any entry in the box by clicking on it and making a new selection from the drop-down box or typing in new information into the edit box, as appropriate.

After checking the parameters in the Properties of Object box, left click once in the Plan View at the *intersection of the grid lines* where you want the column. An I-shaped column should appear at that point in the Plan View. Continue in this manner to place other columns.

Alternatively, draw the remaining columns in one action by "windowing" around the grid intersections. To "window," click the left mouse button above and to the left of your first grid intersection and then, while holding the left mouse button down, drag the mouse until it is below and to the right of the last grid intersection. A selection box similar to that shown in Figure 5-3 should expand around the grid line intersections as the mouse is dragged across the model. Release the left mouse button and the program will draw the column objects at the grid line intersections.

It is a good idea to save your model often. Click the **File menu > Save** command, or the **Save** button, , to save your model.

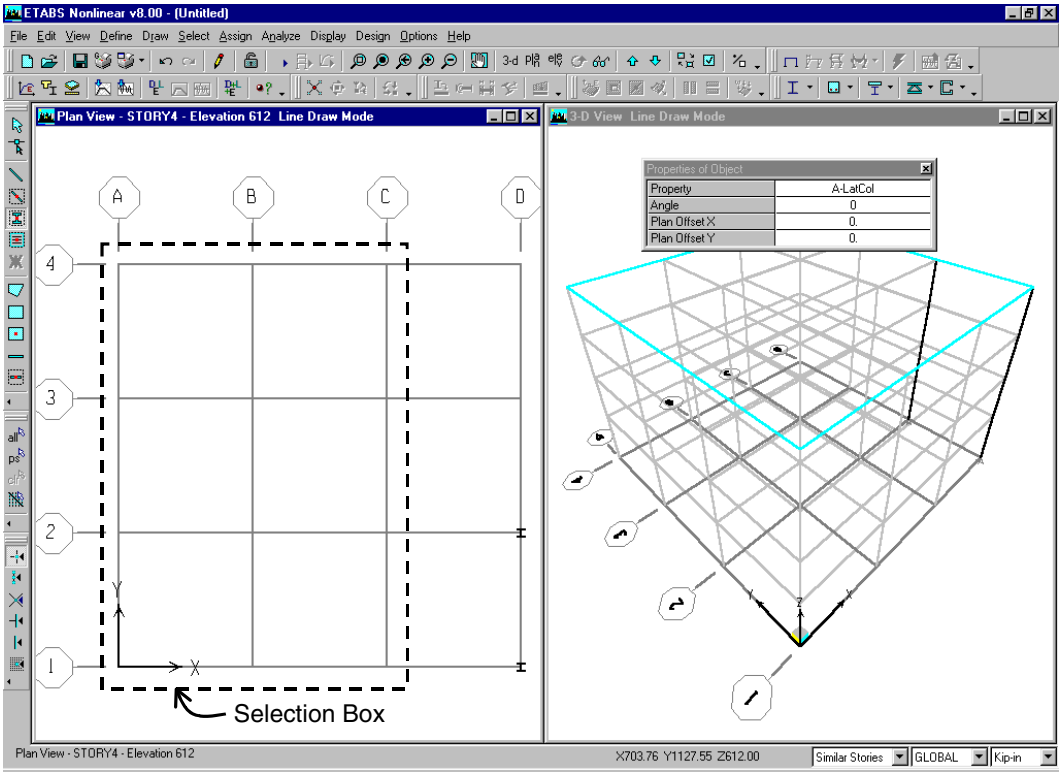



**Figure 5-3 Drawing Column Objects in a Windowed Region**

## Draw Beams

Make sure that the Plan View is active. Click the **Create Lines in Region or at Clicks** button, or the **Draw menu > Draw Line Objects > Create Lines in Region or at Clicks** command. The Properties of Object pop-up box for beams shown in Figure 5-4 will appear.

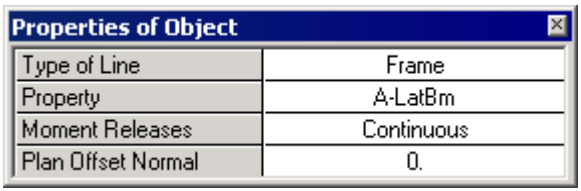



**Figure 5-4 Properties of Object Box for Beams**

As explained previously, the Properties of Object box provides various definition parameters. Change any entry in the box by clicking on it and making a new selection from the drop-down box or typing in new information into the edit box, as appropriate.

After checking the parameters in the Properties of Object box, left click once in the Plan View on a *grid line* where a beam is to be placed. A beam is drawn along the selected grid line. Continue in this manner to place other beams.

Alternatively, draw the remaining beams in one action by windowing around the grid intersections. Windowing is explained in the previous section.

Click the **File menu > Save** command, or the **Save** button, , to save your model.

## Draw Secondary (Infill) Beams

Add secondary or "infill" beams by clicking the the **Create Secondary Beams in Region or at Clicks** button, or the **Draw menu > Draw Lines Objects > Create Secondary Beams in Region or at Clicks** command. Similar to the other drawing operations, a Properties of Object pop-up box will appear that provides the opportunity to define the parameters for the secondary beams.

To place the secondary beams, left click once in the *bay* bounded by grid lines where the secondary beams are to be placed. Similar to columns and the primary beams, secondary beams can be drawn by windowing over the appropriate bays. Note the *Approx. Orientation* parameter to set the span direction.

Click the **File menu > Save** command, or the **Save** button, , to save your model.

## Draw the Floor

Make sure that the Plan View is active. Click the **Draw Areas** button, , or select the **Draw menu > Draw Area Objects > Draw Areas**

command. The Properties of Object pop-up box for areas shown in Figure 5-5 will appear.

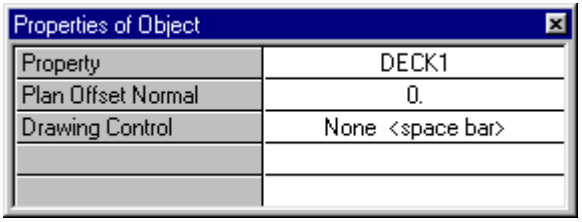



**Figure 5-5 Properties of Object Box for Areas**

Similar to columns and beams, this Properties of Object box provides the opportunity to check and change the parameters for the area. Change any entry in the box by clicking on it and making a new selection from the drop-down box or typing in new information into the edit box, as appropriate.

After checking the parameters in the Properties of Object box, check that the **Snap to Grid Intersections and Points** command is active. This will assist in accurately drawing the area object. This command is active when its associated button is depressed. Alternatively, use the **Draw menu > Snap To > Grid Intersections and Points** command to ensure that this command is active. By default, this command is active.

Left click once at a column to begin the floor/area object at that column. Then, moving around the perimeter of the floor object, click once at other column intersections to draw the outline of the building. Press the Enter key on your keyboard to complete the floor.

If you have made a mistake while drawing this object, click the **Select Object** button, , to change the program from Draw mode to Select mode. Then click the **Edit menu > Undo Area Object Add** command.

To better view the floor addition, click the **Set Building View Options** button . When the Set Building View Options form appears, check the Object Fill check box and the Apply to All Windows check box, as shown in Figure 5-6. Click the **OK** button.

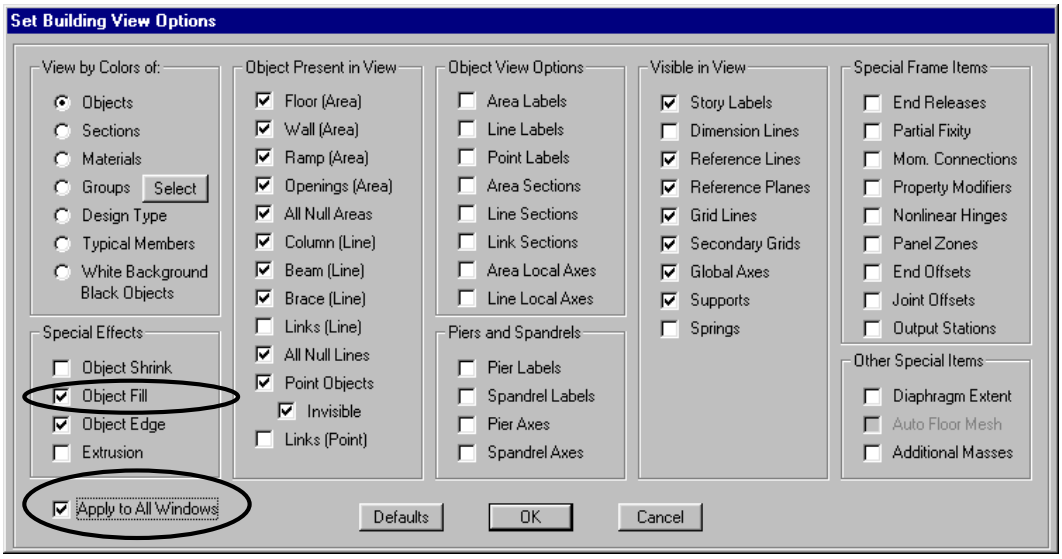



**Figure 5-6 Set Building View Options form**

Click the **File menu > Save** command, or the **Save** button, , to save your model.

Chapter 6

# Select Structural Objects

## Objective

This chapter describes how to select objects in the model.

## Selection Options

The program has three basic methods of selecting objects:

- **Left click:** Left click on an object to select it. If there are multiple objects, one on top of the other, hold down the Ctrl key on your keyboard as you left click on the objects. A form will appear that allows you to specify which object you want to select.
- **Window or "Windowing":** Draw a window around one or more objects to select them. To draw a window around an object, first position your mouse pointer beyond the limits of the object; for example, above and to the left of the object(s) you want to window. Then depress and hold down the left button on your mouse. While keeping the left button

depressed, drag your mouse to a position below and to the right of the object(s) you want to select. Release the left mouse button to complete the selection. Note the following about window selection:

- ✓ As you drag your mouse, a "rubber band window" appears. The rubber band window is a dashed rectangle that changes shape as you drag the mouse. One corner of the rubber band window is at the point where you first depressed the left mouse button. The diagonally opposite corner of the rubber band window is at the current mouse pointer position. Any visible object that is completely inside the rubber band window when you release the left mouse button is selected.

- ✓ As long as you are beyond the limits of the object(s) you want to select, you can start the window at any point; for example, above and to the right, below and to the left, or below and to the right of the object(s) you want to select. In all cases, you would then drag your mouse diagonally across the object(s) you want to select.

- ✓ An *entire* object must lie within the rubber band window for the object to be selected.

***Note about Window Selections in Plan View:*** When selecting by window in a plan view (not a perspective plan view), only the visible objects that lie *fully* in the plane of the plan view are selected. In other words, only the visible point objects, horizontal area objects and horizontal line objects within the select window are selected.

- **Intersecting Line:** Draw a line through one or more objects to select them. To use this selection method, click the **Select menu > using Intersecting Line** command or the **Set Intersecting Line Select Mode** button, . Then position your mouse pointer to one side of the object(s) you want to select. Depress and hold down the left button on your mouse. While keeping the left button depressed, drag your mouse across the object(s) you want to select. Release the left mouse button to complete the selection. Note the following about the intersecting line selection method:

  - ✓ As you drag your mouse a "rubber band line" appears. The rubber band line is a dashed line that changes length and orientation as

you drag the mouse. It extends from the point where you first depressed the left mouse button to the current mouse pointer position. Any visible object that is intersected (crossed) by the rubber band line when you release the left mouse button is selected.

✓ After using this method to make a selection, the program defaults to the window selection mode. Thus, each time you want to use the **Select menu > using Intersecting Line** command, you must use the menu or click **Set Intersecting Line Select Mode** button .



- **Control and Left Click:** Hold down the Ctrl key on the keyboard and left click once on a point, line or area object. A Selection List form similar to the one shown in Figure 6-1 pops up identifying the objects that exist at that location. Select the desired object by moving the mouse pointer over it and left clicking on it.

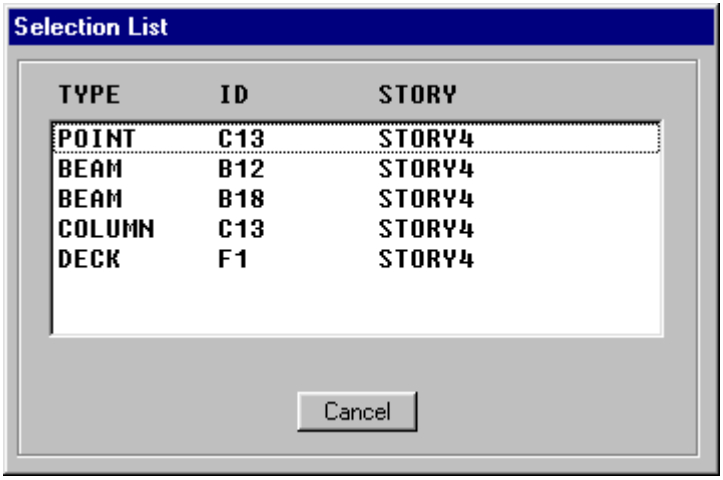



**Figure 6-1 Selection List Form**

# Menu Methods of Selecting Objects

Table 6-1 identifies the submenu commands and related actions accessed using the **Select menu** command.

**Table 6-1 Subcommands of the Select Menu Command**

| Command | Action After Command |
|---|---|
| **on XY Plane** | Click on a single point and all objects (point, line and area) that are in the same XY plane as the selected point are also selected. The object must lie entirely in the associated plane to be selected. |
| **on XZ Plane** | Click on a single point and all objects (point, line and area) that are in the same global XZ plane as the selected point are also selected. The object must lie entirely in the associated plane to be selected. |
| **on YZ Plane** | Click on a single point and all objects (point, line and area) that are in the same global YZ plane as the selected point are also selected. The object must lie entirely in the associated plane to be selected. |
| **by Groups** | Select the name of any collection of objects that has been defined as a group from the Select Group box and that group will be selected. |
| **by Frame Sections** | Select a frame section property name from the Select Sections box and all line objects that have been assigned that frame section property will be selected. |
| **by Wall/Slab/Deck Sections** | Select a wall/slab/deck section property name from the Section Selections box and all area objects that have been assigned that wall/slab/deck section property will be selected. |
| **by Link Properties** | Select a link property name in the Select Properties box and all line objects that have been assigned that link property will be selected. |

**Table 6-1 Subcommands of the Select Menu Command**

| Command | Action After Command |
|---|---|
| **by Line Object Type** | Select a line objects type from the Select Line Object Type box and all line objects of that type will be selected. The choices for the types of line objects are column, beam, brace, null or dimen lines (short for dimension lines). |
| **by Area Object Type** | Select an area objects type from the Select Area Object Type box and all area objects of that type will be selected. The choices for the types of area objects are floor, wall, ramp or null. Note that openings are a subset of null area objects. |
| **by PIER ID** | Select a Pier Label name from the Select Pier IDs box and any Shell/Area that has been assigned that name will be selected. |
| **By Spandrel ID** | Select a Spandrel Label name from the Select Spandrel IDs box and any Shell/Area that has been assigned that name will be selected. |
| **by Story Level** | Select a story level from the Story Level box, and all objects (point, line and area) associated with that story level will be selected. |
| **All** | Selects all objects in the model whether they are visible or not. Be careful using this command. It literally selects ***all*** objects in your model. Also use the **Select All** button to execute this command. |
| **Invert** | Changes the selection such that the currently selected objects are no longer selected and all objects that are not currently selected are selected. |

## Deselect Command

Deselect objects one at a time by left clicking on the selected objects. Alternatively, use the **Select menu > Deselect** command and its subcommands for quicker and more specific deselection actions. This command provides access to subcommands similar to those described in Table 6-1, except that executing the **Select menu > Deselect** command and an associated subcommand deselects rather than selects an object(s). For example, assume that you want to select all of the objects in your model except for those in a particular XZ plane. Do this quickly and easily by first using the **Select menu > Select All** command and then using the **Select menu > Deselect > XZ Plane** command.

## Get Previous Selection Command

The **Select menu > Get Previous Selection** command selects the previously selected object(s). For example, assume you have selected some line objects by clicking on them and assigned frame section properties to them. Use the **Get Previous Selection** command or the **Get Previous Selection** button to select the same line objects and assign something else to them, such as member end releases.

## Clear Selection Command

The **Select menu > Clear Selection** command and its associated **Clear Selection** button clear all currently selected objects. It is an all or nothing command. You cannot selectively clear a portion of a selection using this command.

Chapter 7

# Load the Structural Model

## Objective

This chapter describes how to define structural loads for the model.

## Structural Loads

*Note:*

*An unlimited number of static load cases can be defined in ETABS.*

The program allows the user to define a variety of structural loads, including dead, live, earthquake and wind loads. The user then assigns the loads to various structural objects in the model.

Note that the steel frame, concrete frame, composite floor, steel joist, and concrete shear wall design manuals describe design load combinations in accordance with building codes.

### Define the Static Load Case Name

To add a static load case, click the **Define menu > Static Load Cases** command or click the **Define Static Load Cases** button, , to bring up

the Define Static Load Case Names form. Complete the following actions using that form:

1. Type the name of the load case in the Load edit box. The program does not allow use of duplicate names.

2. Select a load type from the Type drop-down box.

3. Type a self-weight multiplier in the Self-Weight Multiplier edit box (see the explanation about the **self-weight multiplier** that follows).

4. If the load type specified is Quake or Wind, select an option from the Auto Lateral Load drop-down box.

5. Click the **Add New Load** button.

**Note:** If your selected an automatic lateral load in the Auto Lateral Load drop-down box, click the **Modify Lateral Loads** button and review or modify the parameters for the automatic lateral load in the resulting form. Then click the **OK** button to return to the Define Static Load Case Names form.

## SELF-WEIGHT MULTIPLIER

The self-weight of the structure is determined by multiplying the weight-per-unit-volume of each object that has structural properties times the volume of the object. The weight-per-unit-volume is specified in the material properties (search for "material properties" using the **Help menu > Search for Help on** command for more information about material properties and the Material Properties command).

A portion of the self-weight can be applied to any static load case. The self-weight multiplier controls what portion of the self-weight is included in a load case. A self-weight multiplier of 1 includes the full self-weight of the structure in the load case. A self-weight multiplier of 0.5 includes one-half of the self-weight of the structure in the load case.

**Normally a self-weight multiplier of 1 in one static load case only should be specified, usually the dead load load case.** All other static load cases then have self-weight multipliers of zero. Note that a self-weight multiplier of 1 is included in two different load cases, and then those two load cases are combined in a load combination, the results for the load combination are based on an analysis where double the self-weight of the building has been applied as a load.



## Modify an Existing Static Load Case

Use the following procedure and the Define Static Load Case Names form to modify an existing static load case. Recall that the Define Static Load Case Names form is accessed using the **Define menu > Static Load Cases** command or the **Define Static Load Cases** button, :

1. Highlight the existing load case in the Loads area of the form. Note that the data associated with that load case appears in the edit and drop-down boxes at the top of the Loads area.
2. Modify any of the data in the Loads area for the load case.
3. Click the **Modify Load** button. If necessary, click the **Modify Lateral Loads** button to modify the automatic lateral load parameters.

## Delete an Existing Static Load Case

Use the following procedure to delete an existing static load case in the Define Static Load Case Names form. Note that when you delete a static load case, all of the loads that have been assigned to the model as a part of that static load case are also deleted.

- Highlight the existing load case in the Loads area of the form. Note that the data associated with that load case appears in the edit and drop-down boxes at the top of the Loads area.
- Click the **Delete Load** button.

# Assign Structural Loads

The loads cases defined in the previous section can be assigned to points/joints, lines/frames, and areas/shells. The user must first select the element before a load can be assigned to the element. Chapter 6 of this guide describes how to select structural objects.



After the object has been selected, click the **Assign menu** command to access the applicable submenu and assignment options. Table 7-1 identifies the submenus and options.

**TABLE 7-1 Load Commands on the Assign Menu**

*sub menus* / *assignment options*

| Joint/Point Loads | Frame/Line Loads | Shell/Area Loads |
|---|---|---|
| Force | Point | Uniform |
| Ground Displacement | Distributed | Temperature |
| Temperature | Temperature | Wind Pressure Coefficient |

Note that the type of element selected determines which assignment can be made. For example, a wind pressure coefficient assignment cannot be made to a joint/point or frame/line object. Thus, if a joint/point object (e.g., a column in plan) or a frame/line object (e.g., beam) has been selected before clicking the **Assign menu** command, the Shell/Area Loads submenu will not be available.

A form will appear after clicking the **Assign menu** command, the submenu applicable to the type of object, and the desired assignment option. Table 7-2 identifies the forms generated when the various commands are used.

**TABLE 7-2 Input Forms for Load Commands on the Assign Menu**

| Command | Name of Input Form* |
|---|---|
| Joint/Point Loads > | |
| Force | Point Forces |
| Ground Displacement | Ground Displacements |
| Temperature | Point Temperatures |

**TABLE 7-2 Input Forms for Load Commands on the Assign Menu**

| Command | Name of Input Form* |
|---|---|
| Frame/Line Loads > | |
| Point | Frame Point Loads |
| Distributed | Frame Distributed Loads |
| Temperature | Line Object Temperatures |
| Shell/Area Loads > | |
| Uniform | Uniform Surface Loads |
| Temperature | Area Object Temperatures |
| Wind Pressure Coefficient | Wind Pressure Coefficients |



* **Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

Although the form names vary depending on the command used, each form has a drop-down menu that allows the user to select the load case to be assigned. Logically, the available load cases vary depending on the type of assignment. The forms also include other object/assignment-specific input fields that enable the user to refine the load assignment.

Search for "static loads" using the **Help menu > Search for Help on** command for more information about the load commands.

# Assign/Change Properties

## Objective

This chapter describes how to assign or change the properties of structural elements in the model.

## Properties

In creating the model, the user draws point, line, and area objects. To enable analysis and design, those objects must be assigned properties, such as material properties, frame sections, wall/slab/deck sections, link properties, and loads, among others. Note that the assign menu lists the various properties that can be assigned. Also note that the assignment of loads was explained in Chapter 7 of this guide.

As shown in Table 8-1, the types of assignments available depend on the type of object. Assignments also depend on the type of design (e.g., steel versus concrete versus composite design).

**TABLE 8-1 Possible Assignments to Objects by Object Type**

| Object | Assignment Option | Name of Input Form* |
|---|---|---|
| **Joint/Points** | Rigid Diaphragms | Assign Diaphragm |
| | Panel Zones | Assign Panel Zone |
| | Restraints (Supports) | Assign Restraints |
| | Point Springs | Assign Springs |
| | Link Properties | Assign NLLink Properties |
| | Additional Point Masses | Assign Masses |
| **Frame/Lines** | Frame Sections | Assign Frame Properties |
| | Frame Releases/Partial Fixity | Assign Frame Releases |
| | Moment Frame Beam Type | Special Moment Beams |
| | End (Length) Offsets | Frame End Length Offsets |
| | Insertion Point | Frame Insertion Point |
| | Frame Output Stations | Assign Output Station Spacing |
| | Local Axes | Axis Orientation |
| | Frame Property Modifiers | Analysis Property Modification Factor |
| | Frame Line Types | Assign Frame Line Type |
| | Link Properties | Assign NLLink Properties |
| | Frame NonLinear Hinges | Assign Frame Hinges (Pushover) |
| | Pier Labels | Pier Names |
| | Spandrel Labels | Spandrel Names |
| | Line Springs | Assign Spring |
| | Additional Line Masses | Assign Mass |
| | Automatic Frame Subdivide | ***Subdivide, Don't Subdivide, Cancel* |
| | Use Line for Floor Meshing | ***Yes, No, Cancel* |
| **Shell/Areas** | Wall/Slab/Deck Sections | Assign Wall/Slab/Deck Section |
| | Openings | Assign Openings |
| | Rigid Diaphragms | Assign Diaphragms |
| | Local Axes | Assign Local Axis |
| | Shell Stiffness Modifiers | Analysis Stiffness Modification Factors |
| | Pier Labels | Pier Names |
| | Spandrel Labels | Spandrel Names |
| | Area Springs | Assign Spring |
| | Additional Area Masses | Assign Mass |
| | Area Object Mesh Options | Area Object Auto Mesh Options |
| | Auto Line Constraint | Auto Line Constraint Options |

*** Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

**Not a form; possible input parameters

View the assignment made to point, line, and area objects by *right* clicking on the object. The appropriate Point Information, Line Information, or Area Information form will display. Click on the Assignment tab.

In each case, select an object before executing the desired assignment command (e.g., select a line object before using the **Assign menu > Frame/Lines > Frame Sections** command). As explained in Chapter 6 of this guide, using the Ctrl key and left clicking on a location in the model can simplify the process of selecting objects when multiple objects may be present at the same location or if selecting objects is new to the user and seems challenging.



The availability of commands depends on the type of object selected. The input forms include object/assignment-specific input fields that enable refinement of the assignment. Modifications to the assignments can be made by accessing the input forms using the appropriate Assign menu command.

The forms include **OK** and **Cancel** buttons that can be used to accept or delete changes made to the forms.

Note that the combination of the type of object, name of the command and name of the input form provides an indication of what can be achieved by using a particular command.

## Auto Select Section List

ETABS's Auto Select Section List feature helps to control the time required to develop the model as well as to enhance the design process.

An auto select selection list is simply a list of sections; for example, W18X35, W18X40, W21X44, W21X50 and W24X55. Auto select section lists can be assigned to frame members. When an auto select selection list is assigned to a frame member, the program can automatically select the most economical, adequate section from the auto select section list when it is designing the member.

The program has several built-in auto select section lists. However, the user can also develop a tailored list using the following steps:

1. Click the **Define menu > Frame Sections** command, which will display the Define Frame Properties form shown in Figure 8-1.

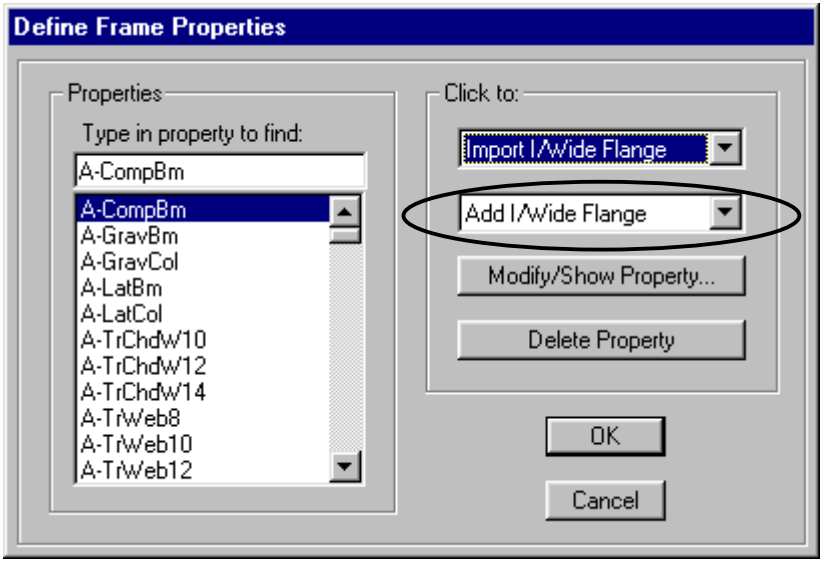



**Figure 8-1: The Define Frame Properties form**

2. Click the drop-down box that reads "*Add I/Wide Flange"* in the Click To area of the Define Frame Properties form. Scroll down the resulting list of potential Add sections to locate *Add Auto Select List*. Double click on it. The Auto Selection Sections form shown in Figure 8-2 appears.

3. Type a name for the list in the Auto Section Name edit box. Any name can be used. For the purposes of this description, the new Auto Select Section List is **AUTOLATBM.**

4. Scroll down the list of beam sections in the List of Sections to find the beams to be included in the list. Click once on them to highlight them. Note that the standard Windows methods for selecting items in a list can be used (e.g., clicking on a beam and then pressing the shift key on the key board before selecting another beam will highlight all beams between the two selected beams).

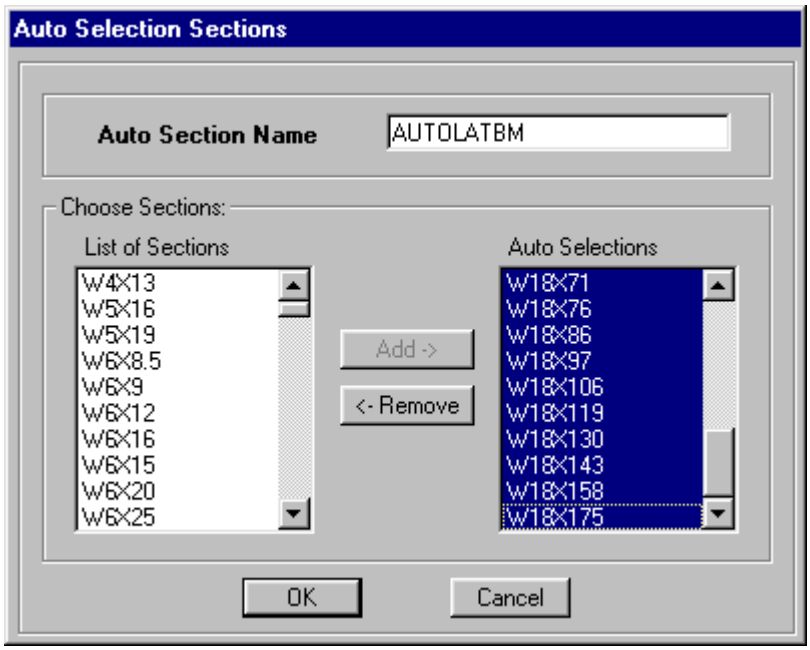



**Figure 8-2: Auto Select Sections form**

5. Click the **Add** button to add the selected beams to the Auto Selections list on the right side of the form.

6. Click the **OK** button and then click the **OK** button in the Define Frame Properties form to accept the definition of a new Auto Select Section List called AUTOLATBM.

## Assign the AUTOLATBM Auto Select Section List

The AUTOLATBM Auto Select Section list created as described in the previous section consists of various sections that can be assigned to a frame element. Thus, in making the assignment, the user should ***not*** select a point or area object in the model, or click the Joint/Point or Shell/Area commands on the Assign menu.

Rather, the user should select a frame/line object (e.g., a beam) and then click the **Assign menu > Frame/Line > Frame Section** command or the **Assign Frame Section** button, . This will bring up the Assign Frame

Properties form. In the Properties area of that form, scroll down the list of properties to locate and highlight the name of the Auto Select Section List to be assigned; AUTOLATBM in this example. Note that when a name is highlighted in the list, the name also appears in the edit box at the top of the list. Click the **OK** button and the assignment of the Auto Select Section List named AUTOLATBM is complete.

## Make an Assignment as the Object is Drawn



An Auto Select Section List can also be assigned when the frame/line object is being drawn on the model. Using this method, select the desired Auto Select Section list by name from the drop-down box of the Properties of Object Box that appears when a drawing tool is selected. Use of the drawing tools is described in Chapter 5 of this guide along with figures showing the Properties of Object boxes for point, line, and area objects.

## Check the Sections in an Auto Select Section List

As indicated previously, several Auto Select Section Lists are built into the program. To review the sections included in any Auto Select Section Lists, whether built in or user-specified, complete the following steps:

1. Click the **Define menu > Frame Sections** command or click the **Define Frame Sections** button . The Define Frame Properties form will appear.

2. Highlight the name of the Auto Select Section List to be checked in the Properties drop-down list.

3. Click the **Modify/Show Property** button. The Auto Selection Sections form appears; the sections included in the selected auto select section list are listed in the Auto Selections area of the form.

4. Click the **Cancel** button to close the form.

# Edit the Model Geometry

## Objective

This chapter describes how to edit the model quickly and easily while maintaining model integrity.

## Editing Options

During the course of creating the model, the model may require editing. Table 9-1 identifies the various edit command available in ETABS. Some are familiar Windows commands.

In most cases, first select the point, line, or area object, then click the appropriate menu or button. In some cases, the action will be immediate (e.g., the Undo or Redo commands). In other cases, a form will appear that allows the user to specify how the object is to be edited (e.g., the **Edit menu > Align Points/Lines/Edges** command brings up the Align Points/Lines/Edges form, which allows the user to align *points* to the x, y, z coordinate or to the nearest point, or to trim or extend *lines*). In other

cases, the command is a toggle that, when enabled, will affect subsequent actions. Note that the type of commands and options available depend on the type of object being edited.

**TABLE 9-1 Edit Commands in ETABS**

| Command | Action | Immediate/ Form* Used/ Toggle |
|---|---|---|
| **Undo and Redo** | Undo deletes the last performed action. Redo restores the last step that was undone. | Immediate |
| **Cut, Copy and Paste** | Generally similar to the standard cut, copy and paste Windows commands, with some ETABS specific behaviors. Only active in plan or plan perspective view. | Immediate |
| **Delete** | Delete deletes the selected object(s) and all of its assignments (loads, properties, supports and the like). | Immediate |
| **Add to Model from Template** | Add two-dimensional and three-dimensional frames to the model. | 2-D or 3-D Forms that access other forms |
| **Replicate** | Replicate replicates one or more objects and most of the object's assignments. Note that replicated objects will ***not*** replace objects already placed at a location. | Replicate Form that accesses options form |
| **Edit Grid**<br>> Edit Grid Data | Edit Grid Data edits the coordinate system. Resulting form allows user to select a previously defined grid system; define a new system; add a copy of an existing system; show/modify an existing system; and delete an existing system. | Coordinate Systems Form that accesses definition forms |
| > Add Grid at Selected Points | Adds grid lines at selected points. | Add Grid Lines at Selected Point Form |
| > Glue Points to Grid Lines | "Glues" point objects that lie directly on grid lines to those grid lines. When a point object is glued | Toggle |

**TABLE 9-1 Edit Commands in ETABS**

| Command | Action | Immediate/ Form* Used/ Toggle |
|---|---|---|
| | to a grid line and the grid line is moved, the point object moves with the grid line. Line and area objects that are attached to the point object when it is moved remain attached to the point object and move or resize as appropriate. | |
| > Lock OnScreen Grid System Edit | Allows users to lock out the ability to move grid lines graphically on-screen using the **Reshape Object** command. | Toggle |
| **Edit Story Data**<br>> Edit Story | Edit Story allows changes to story labels, heights, master story designation, Similar To designation, splice point and splice height. Allows changes to base story elevation. | Story Data Form |
| > Insert Story | Inserts new story into the model. Allows the user to define story name, story height, number of stories to insert, location/placement of inserted story(ies) and if the story is to replicate from another story (i.e., be a copy of another story, including properties). | Insert New Story Form |
| > Delete Story | Allows user to select and delete a story from the model. | Select Story to Delete Form |
| **Edit Reference Planes** | Allows the user to create, modify and delete reference planes. Reference planes are horizontal planes at user-specified Z-ordinates. Reference planes provide a horizontal plane that can be used when drawing objects in elevation views to snap objects into place (for more information search for “snap to” using the Help menu). | Edit Reference Planes Form |

**TABLE 9-1 Edit Commands in ETABS**

| Command | Action | Immediate/ Form* Used/ Toggle |
|---|---|---|
| **Edit Reference Lines** | Allows the user to create, modify and delete reference lines. Reference lines are vertical lines at user-specified global X and Y coordinates. Reference lines can be used when drawing objects in elevation or plan views to snap objects into place (for more information search for "snap to" using the Help menu) . | Edit Reference Lines Form |
| **Merging Points** | Merge Points merges points within the tolerance distance of the selected point. The user specifies the tolerance distance on the Merge Selected Points Form. | Merge Selected Points Form |
| **Align Points/Lines/ Edges** | Align Points/Lines/Edges helps the user align objects in the model. Search for "edit points lines edges" using the **Help menu > Search for Help on** command for important notes about using this command. | Align Selected Points/Lines/ Edges Form |
| **Move Points/Lines/ Areas** | Move Points/Lines/Areas helps the user move objects in the model. Search for "move points lines edges" using the **Help menu > Search for Help on** command for more information. | Move Points/Lines/ Area Form |
| **Expand/Shrink Areas** | Expand/Shrink Areas expands or shrinks an area object using a user-specified offset value. | Expand/Shrink Areas Form |
| **Merge Areas** | Merge Areas merges two area objects that have a common edge or overlap into one area object. | Immediate *Can use Undo* |
| **Mesh Areas** | Mesh Areas meshes (i.e., divides) selected areas using user-specified meshing options. Search for "mesh" using the **Help menu > Search for Help on** command for more information. | Mesh Selected Areas Form |
| **Split Area Edges** | Adds point objects at the mid-point of each edge of an area object. | Immediate *Can use Undo* |

**TABLE 9-1 Edit Commands in ETABS**

| Command | Action | Immediate/ Form* Used/ Toggle |
|---|---|---|
| **Join Lines** | Join Lines joins two or more collinear line objects with common end points and the same type of property into a single line object. | Immediate *Can use Undo* |
| **Divide Lines** | Divide Lines divides a line object into multiple line objects. | Divide Selected Lines Form |
| **Extrude Point to Lines** | Creates line objects from points. Options are available for linear or radial extrusion. This feature is especially suited to creating beams/columns from point/nodes. | Extrude Point to Lines Form |
| **Extrude Lines to Areas** | Extrude Lines to Area creates area objects from lines. Options are available for linear or radial extrusion. This feature is especially suited to creating area objects from beams. | Extrude Lines to Areas Form |
| **Auto Relabel All** | Relabels all objects of the current model. Command cannot be undone. Use this command after model creation is complete to get optimum labeling for the model. | Warning message *Cannot use Undo* |
| **Nudge** | Works with Ctrl and arrow keys to move objects. Allows the user to select objects and move them a predefined distance. For more information, search for “nudge” using the Help menu. | Immediate |

* **Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

# Analyze the Model

## Objective

This chapter describes how to analyze the model.

## Model Analysis

To run the analysis, click the **Analyze menu > Run Analysis** command or the **Run Analysis** button, , and click the **Run** button on the Run Options form.

The program will display an "Analyzing, Please Wait" window. Data will scroll in this window as the program runs the analysis. After the analysis has been completed, the program performs a few more “book-keeping actions” that are evident on the status bar in the bottom left-hand corner of the ETABS window.

When the entire analysis process has been completed, the model automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the Lock/Unlock Model button, , appears depressed. Locking the model prevents any changes to the model that would invalidate the analysis results.

Chapter 11

11

# Design

## Objective

This chapter describes design using the ETABS design postprocessors.

## Design the Structure

The ETABS design postprocessors include the following:

- Steel Frame Design
- Concrete Frame Design
- Composite Beam Design
- Steel Joist Design
- Shear Wall Design

To perform the design, first run the analysis (described in Chapter 10), then click the Design menu and select the appropriate design from the drop-down menu. The type of design available depends on the type of members used in the model. That is, the user cannot complete a shear wall design if no shear walls have been included in the model.

Similarly, the commands used to execute a design depend on the type of design to be performed. However, each design has commands to address the following:

- Review and/or select design load combinations.
- Review and/or select overwrites.
- Start the design or check of the structure.
- Perform interactive design.
- Display input and output design information on the model.



Generally, the sequence for using commands is indicated by their availability. In other words, some commands must be used before other commands become available. This helps the user step through the design process. (Search for "process" using the Help menu to access more information about design processes and the sequence of commands.) Table 11-1 identifies the commands that are used to start design depending on the desired design process.

**TABLE 11-1 Start Design Commands**

| Design Process | Command that Starts Design |
|---|---|
| Steel Frame Design | Start Design/Check of Structure |
| Concrete Frame Design | Start Design/Check of Structure |
| Composite Beam Design | Start Design using Similarity or<br>Start Design Without Similarity |
| Steel Joist Design | Start Design using Similarity or<br>Start Design Without Similarity |
| Shear Wall Design | Start Design/Check of Structure |

It is important to understand that design in ETABS is an iterative process. That is, the user should run the analysis and then perform the design and be prepared to run the analysis again and perform the design again. It may be necessary to repeat this process several times before the design is complete. The objective is to have the analysis sections match the design sections. The program will report any differences. The user should repeat the analysis/start design process until the analysis and design sections match (i.e., the program does not display an error message).

Tables 11-2 through 11-6 summarize the commands used in each type of design process.



**Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

**TABLE 11-2 Steel Frame Design Commands**

| Command | Action | Form |
|---|---|---|
| **Select Design Group** | Designates that a group is to be used as a design group. Works only when auto select sections have been assigned to frame objects. All frame objects in the group will be given the same design section. | Steel Frame Design Group Selection Form |
| **Select Design Combo** | Allows review of the default steel frame design load combinations defined by the program, or designation of user-specified design load combinations. Facilitates review or modification of loads during design. | Design Load Combinations Selection Form |
| **View/Review Overwrites** | Allows review of overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the frame elements to which they are specifically assigned. | Overwrites Form |
| **Set Lateral Displacement Targets** | Specifies displacement targets, in any direction, for various load cases. | Lateral Displacement Targets Form |

**TABLE 11-2 Steel Frame Design Commands**

| Command | Action | Form |
|---|---|---|
| **Set Time Period Targets** | Specifies time period targets for seismic analysis. | Time Period Targets Form |
| **Start Design/Check of Structure** | Initiates design process. If frame elements have been selected before this command is clicked, only the selected frame elements will be designed. A building analysis must precede use of this command. | Immediate, no form used |
| **Interactive Steel Frame Design** | Allows the user to review the design results for any frame element and then to interactively change the design overwrites and immediately view the results. | No form; results are displayed onscreen. |
| **Display Design Info** | Allows review of some of the results of the steel frame design directly on the program model. Examples of results that can be displayed include design sections, unbraced lengths, effective length factors, allowable stresses, and stress ratio information. | Display Design Results Form |
| **Make Auto Select Section Null** | Removes auto select section lists from selected frame elements. Typically used near the end of the iterative design process so that the final design iteration is performed using the actual frame sections assigned, not auto select sections. Only works on a user-specified selection. | Warning message *Cannot use Undo* |
| **Change Design Section** | Allows the user to change the design section property assigned to one or more frame elements and then rerun the design without first rerunning the analysis. Only works on a user-specified selection. | Select Sections Form |
| **Reset Design Section to Last Analysis** | Sets the design section for one or more frame elements back to the last used analysis section. Only works on a user-specified selection. | Immediate *Cannot use Undo* |
| **Verify Analysis vs Design Section** | Verifies that the last used analysis section and the current design section are the same for all steel frame elements in the model. | Immediate |

**TABLE 11-2 Steel Frame Design Commands**

| Command | Action | Form |
|---|---|---|
| **Verify All Members Passed** | Reports if structural members have passed the stress/capacity check. An analysis and a design/check of the structure must be completed before this command. | Immediate |
| **Reset All Steel Overwrites** | Resets the overwrites for all frame sections with the Steel Frame design procedure to their default values. | Warning message *Cannot use Undo* |
| **Delete Steel Design Results** | Deletes all of the steel frame design results but not the current design section (i.e., next analysis section). | Immediate *Cannot use Undo* |



**TABLE 11-3 Concrete Frame Design Commands**

| Command | Action | Form |
|---|---|---|
| **Select Design Combo** | Allows review of the default concrete frame design load combinations defined by the program, or designation of user-specified design load combinations. Facilitates review or modification of loads during design. | Design Load Combinations Selection Form |
| **View/Review Overwrites** | Allows review of overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the frame elements to which they are specifically assigned. | Overwrites Form |
| **Start Design/Check of Structure** | Initiates design process. If frame elements have been selected before this command is used, only the selected frame elements will be designed. A building analysis must precede use of this command. | Immediate, no form used |
| **Interactive Concrete Frame Design** | Allows the user to review the design results for any frame element and then to interactively change the design overwrites and immediately view the results. | No form; results are displayed onscreen. |

**TABLE 11-3 Concrete Frame Design Commands**

| Command | Action | Form |
|---|---|---|
| **Display Design Info** | Allows review of some of the results of the concrete frame design directly on the program model. Examples of results that can be displayed include design sections, unbraced lengths and longitudinal reinforcing. | Display Design Results Form |
| **Change Design Section** | Allows the user to change the design section property assigned to one or more frame sections and then rerun the design without first rerunning the analysis. Only works on a user-specified selection. | Select Sections Form |
| **Reset Design Section to Last Analysis** | Sets the design section for one or more frame elements back to the last used analysis section. Only works on a user-specified selection. | Immediate *Cannot use Undo* |
| **Verify Analysis vs Design Section** | Verifies that the last used analysis section and the current design section are the same for all concrete frame elements in the model. | Immediate |
| **Reset All Concrete Overwrites** | Resets the overwrites for all frame sections with the Concrete Frame design procedure to their default values. | Warning message *Cannot use Undo* |
| **Delete Concrete Design Results** | Deletes all of the concrete frame design results but not the current design section (i.e., next analysis section). | Immediate *Cannot use Undo* |

**TABLE 11-4 Composite Beam Design Commands**

| Command | Action | Form |
|---|---|---|
| **Select Design Group** | Designates that a group is to be used as a design group. Works only when auto select sections have been assigned to frame objects. When grouped, all beams in the group are given the same beam size, but the shear connectors and camber may be different. | Composite Design Group Selection Form |

**TABLE 11-4 Composite Beam Design Commands**

| Command | Action | Form |
|---|---|---|
| **Select Design Combo** | Allows review of the default composite frame design load combinations defined by the program, or designation of user-specified design load combinations. Facilitates review or modification of loads during design. Note that separate design load combinations are specified for construction loading, final loading considering strength, and final loading considering deflection. | Design Load Combinations Selection Form |
| **View/Review Overwrites** | Allows review of overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the composite beams to which they are specifically assigned. | Overwrites Form |
| **Start Design using Similarity** | Assumes that if a composite beam is located at a story designated as similar to a master story, that composite beam has the same composite beam size as the composite beams of the master story (set a story Similar To a master story in the Story Data; see the **Edit menu > Edit Story Data > Edit Story** command). | Immediate, no form used |
| **Start Design without Similarity** | Excludes the similarity features described in the description of the Start Design using Similarity command. Can be applied to a user-specified selection only. ***ALWAYS*** use this command for final design. | Immediate, no form used |
| **Interactive Composite Beam Design** | Allows the user to review the design results for any composite beam and then to interactively change the design overwrites and immediately view the results. | No form; results are displayed onscreen. |
| **Display Design Info** | Allows review of some of the results of the composite beam design directly on the program model. Examples of results that can be displayed include beam labels and associated design group names; design sections together with connector layout, camber and end reactions; and stress ratio information. | Display Design Results Form |

**TABLE 11-4 Composite Beam Design Commands**

| Command | Action | Form |
|---|---|---|
| **Make Auto Select Section Null** | Removes auto select section lists from selected beams. Typically used near the end of the iterative design process so that the final design iteration is performed using the actual beam sections assigned, not auto select sections. Only works on a user-specified selection. | Warning message *Cannot use Undo* |
| **Change Design Section** | Allows the user to change the design section property assigned to one or more beams and then rerun the design without first rerunning the analysis. Only works on a user-specified selection. | Select Sections Form |
| **Reset Design Section to Last Analysis** | Sets the design section for one or more beams back to the last used analysis section. Only works on a user-specified selection. | Immediate *Cannot use Undo* |
| **Verify Analysis vs Design Section** | Verifies that the last used analysis section and the current design section are the same for all composite beams in the model. | Immediate |
| **Verify All Members Passed** | Reports if structural members have passed the stress/capacity check. An analysis and a design/check of the structure must be completed before this command. | Immediate |
| **Reset All Composite Beam Overwrites** | Resets the overwrites for all composite beams with the Composite Beam design procedure to their default values. | Warning message *Cannot use Undo* |
| **Delete Composite Beam Results** | Deletes all of the composite beam design results but not the current design section (i.e., next analysis section). | Immediate *Cannot use Undo* |

**TABLE 11-5 Steel Joist Design Commands**

| Command | Action | Form |
|---|---|---|
| **Select Design Group** | Designates that a group is to be used as a design group. Works only when auto select sections have been assigned to the joints. When grouped, all objects in the group are given the same joist size. | Composite Design Group Selection Form |

**TABLE 11-5 Steel Joist Design Commands**

| Command | Action | Form |
|---|---|---|
| **Select Design Combo** | Allows review of the default steel joist design load combinations defined by the program, or designation of user-specified design load combinations. Facilitates review or modification of loads during design. | Design Load Combinations Selection Form |
| **View/Review Overwrites** | Allows review of overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the steel joists to which they are specifically assigned. | Overwrites Form |
| **Start Design using Similarity** | Assumes that if a steel joist is located at a story designated as similar to a master story, that steel joist has the same joist size as the steel joists of the master story (set a story Similar To a master story in the Story Data; see the **Edit menu > Edit Story Data > Edit Story** command). | Immediate, no form used |
| **Start Design Without Similarity** | Excludes the similarity features described in the description of the Start Design using Similarity command. Can be applied to a user-specified selection only. ***ALWAYS*** use this command for final design. | Immediate, no form used |
| **Interactive Steel Joist Design** | Allows the user to review the design results for any steel joist and then to interactively change the design overwrites and immediately view the results. | No form; results are displayed onscreen. |
| **Display Design Info** | Allows review of some of the results of the steel joist design directly on the program model. Examples of results that can be displayed include joist labels and associated design group names; design sections together with end reactions; and design ratio information. | Display Design Results Form |

**TABLE 11-5 Steel Joist Design Commands**

| Command | Action | Form |
|---|---|---|
| **Make Auto Select Section Null** | Removes auto select section lists from selected joists. Typically used near the end of the iterative design process so that the final design iteration is performed using the actual joists sections assigned, not auto select sections. Only works on a user-specified selection. | Warning message *Cannot use Undo* |
| **Change Design Section** | Allows the user to change the design section property assigned to one or more joists and then rerun the design without first rerunning the analysis. Only works on a user-specified selection. | Select Sections Form |
| **Verify Analysis vs Design Section** | Verifies that the last used analysis section and the current design section are the same for all steel joists in the model. | Immediate |
| **Verify All Members Passed** | Reports if all steel joists have passed the stress/capacity check. An analysis and a design/check of the structure must be completed before this command. | Immediate |
| **Reset All Steel Joist Overwrites** | Resets the overwrites for all steel joists with the Steel Joist design procedure to their default values. | Warning message *Cannot use Undo* |
| **Delete Steel Joist Results** | Deletes all of the steel joist design results but not the current design section (i.e., next analysis section). | Immediate *Cannot use Undo* |

**TABLE 11-6 Shear Wall Design Commands**

| Command | Action | Form |
|---|---|---|
| **Select Design Combo** | Allows review of the default shear wall design load combinations defined by the program, or designation of user-specified design load combinations. Facilitates review or modification of loads during design. | Design Load Combinations Selection Form |

**TABLE 11-6 Shear Wall Design Commands**

| Command | Action | Form |
|---|---|---|
| **View/Review Pier Overwrites** | Allows review of pier overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the piers to which they are specifically assigned. | Overwrites Form |
| **View/Review Spandrel Overwrites** | Allows review of spandrel overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the spandrels to which they are specifically assigned. | Overwrites Form |
| **Define General Pier Sections** | Allows the user to define a pier section using the Section Designer utility. | Define General Pier Sections Form |
| **Assign Pier Sections Type** | Allows the user to assign a pier one of three section types. | Assign Pier Sections Form |
| **Start Design/Check of Structure** | Initiates design process. If piers or spandrels have been selected before this command is used, only the selected piers or spandrels will be designed. A building analysis must precede use of this command. | Immediate, no form used |
| **Interactive Wall Design** | Allows the user to review the design results for any piers or spandrels and then to interactively change the design overwrites and immediately view the results. | No form; results are displayed onscreen. |
| **Display Design Info** | Allows review of some of the results of the shear wall design directly on the program model. Examples of results that can be displayed include reinforcing requirements, capacity ratios and boundary element requirements. | Display Design Results Form |
| **Reset All Pier/ Spandrel Overwrites** | Resets the overwrites for all piers or spandrels to their default values. | Warning message *Cannot use Undo* |
| **Delete Wall Design Results** | Deletes all of the shear wall results. | Immediate *Cannot use Undo* |

Chapter 12

12

# Graphical Displays

## Objective

This chapter describes how to display analysis results graphically.

## Obtain Basic Graphical Displays

Analysis results can be displayed graphically after the analysis has been run. To display results, click the Display menu and select the type of display desired. Table 12-1 identifies the display options.

**TABLE 12-1 Display Menu Options**

| Command | Action | Form* |
|---|---|---|
| **Show Undeformed Shape** | Show Undeformed Shape plots the undeformed shape onscreen. | No form; results are displayed. |
| **Show Loads** | | |
| > Joint/Point | Displays loads assigned to Points. | Show Joint/Point Loads Form |
| > Frame/Line | Displays loads assigned to Lines. | Show Frame/Line Loads Form |
| > Shell/Area | Displays loads assigned to Areas. | Show Shell/Area Loads Form |
| **Set Input Table Mode** | Set Input Table Mode provides the user with the opportunity to complete an onscreen review of the input parameters used in building the model. | Data Tables Input Form |
| **Show Deformed Shape** | Display Static Deformed Shape plots a deformed shape onscreen based on user-specified loads. This plot can be animated. | Deformed Shape Form |
| **Show Mode Shape** | Show Mode Shape plots a deformed shape onscreen based on user-specified modes. This plot can be animated. | Mode Shape Form |
| **Show Member Forces/ Stress Diagram** | | |
| > Support/Spring Reactions | Displays support and spring reactions onscreen based on user-specified loads. | Point Reaction Forces Form |
| > Frame/Pier/ Spandrel Forces | Displays column, beam, brace, pier and spandrel forces onscreen based on user-specified loads. | Member Force Diagram for Frames Form |
| > Shell Stresses/ Forces | Displays internal shell element forces and stresses onscreen based on user-specified loads. | Element Force/Stress Contours for Shells Form |
| > Link Forces | Displays link forces onscreen based on user-specified loads. | Member Force Diagram Form |

**TABLE 12-1 Display Menu Options**

| Command | Action | Form* |
|---|---|---|
| **Show Energy/Virtual Work Diagram** | Show Energy/Virtual Work Diagram displays energy/virtual work diagrams that can be used as an aid to determine which elements should be stiffened to most efficiently control the lateral displacements of the structure. User defines forces and displacements | Energy/Virtual Work Diagrams |
| **Show Response Spectrum Curves** | Show Response Spectrum Curves plots various response spectra ***after a time history analysis has been run.*** | Response Spectrum Generation Form |
| **Show Time History Traces** | Show Time History Traces plots various time history curves based on user-specified data after a time history analysis has been run. | Time History Display Definition Form |
| **Show Static Pushover Curve** | Show Static Pushover Curve displays various pushover curves based on user specified data after a static nonlinear analysis has been run. | Pushover Curve Form |
| **Set Output Table Mode** | Set Output Table Mode allows the user to select the type of information to include in the output tables. | Display Output Tables Form |

* **Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

Chapter 13

13

# Generate Results

## Objective

This chapter describes how to generate analysis and design results that can be printed to a printer or to a file for sharing with other programs.

## Analysis and Design Results

Analysis and design results can be printed to a printer or a file using the File menu commands. Table 13-1 identifies the print commands.

**TABLE 13-1 File Menu Print Options**

| Command | Action | Form* |
|---|---|---|
| **Print Setup** | Allows the user to specify the paper size and orientation of the page. | Print Page Setup Form |
| **Print Preview for Graphics** | Provides a snap shot of how the file will print in graphical format. | N/A |
| **Print Graphics** | Print Graphics prints whatever graphics are displayed in the active window to the printer that is currently specified as active. | N/A |
| **Print Tables** | | |
| > Input | Prints tables of analysis input data to a printer or to a text file. | Print Input Tables Form |
| > Analysis Output | Prints tables of analysis output data to a printer or to a text file. | Print Output Tables Form |
| > Print Design Tables | Prints tables of design output data to a printer or to a text file based on which design the user selects: Steel Frame, Concrete Frame, Composite Beam, Steel Joist, or Shear Wall. | Print Design Tables Form |

* **Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.